

i-feature

intelligent machining systems

Installation and Operation Manual

Table of Contents:

1. System requirements and notices
2. I-feature installation
3. Getting started
4. Uninstalling i-feature

Appendices:

Appendix 1. Detailed description of geometry parameters input in the “i-feature Planar Mill” operations

1. System requirements and notices



To install and run i-feature, installed version (v. 8.5, 9.0, 10.0 or 11.0 including update) of Siemens NX is required!



To install/uninstall i-feature, admin rights are required!

If you have any problems or questions, please e-mail technical support service to: support@i-feature.net

2. I-feature installation

To install i-feature go to “Product” section on <http://i-feature.net> and click “Download”. Then click the link “Terms of Service” to read and agree with the Terms of Service. After accepting the terms, click “Place an Order/Checkout”. In the window that opens, enter your e-mail and click “Next”. In the next window click “Finish” and you will see complete information on the order and download link to the product. This link will be also duplicated on the specified e-mail.



Before installing i-feature all windows of Siemens NX must be closed, if any!

After downloading the distribution, run its executable file “i-feature for NX Setup.exe”. In the window that opens (Figure 2.1), enter your e-mail, specified earlier. The i-feature installation has been successfully completed.



Figure. 2.1. The i-feature Setup window

After that, open Siemens NX program. In the application for mechanical treatment you can see now i-feature entry button. It is located next to the “Create Operations” button. A new group called “i-feature Operations” is also added to the menu. You can also place the “i-feature Create Operations” button anywhere you want.

3. Getting started

Before getting started i-feature, you must set a material in the Geometry of workpiece (Figure 3.1).

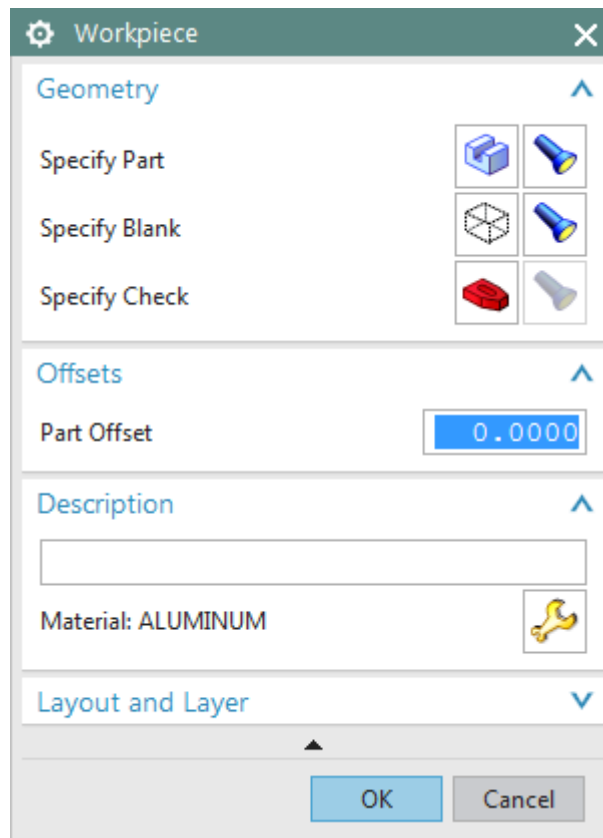


Figure 3.1. Workpiece settings

There are three groups of workpiece in i-feature:

- **MAT0_0002 : CARBON STEEL** (Carbon and low-alloy steel of general purpose)
- **MAT0_0153 : STAINLESS STEEL** (Stainless and high-alloy steel, heat-resistant steel and alloys)
- **MAT0_0266 : ALUMINIUM** (Aluminum alloys and free-cutting material materials)

I-feature users can work in the following dialog boxes:

- the “i-feature Create Operations” window (Figure 3.2)
- the “i-feature Planar Mill” window (Figure. 3.3)
- the “Boundary Geometry” and “Create Boundary” windows to set and modify the machining geometry (Figure 3.4)

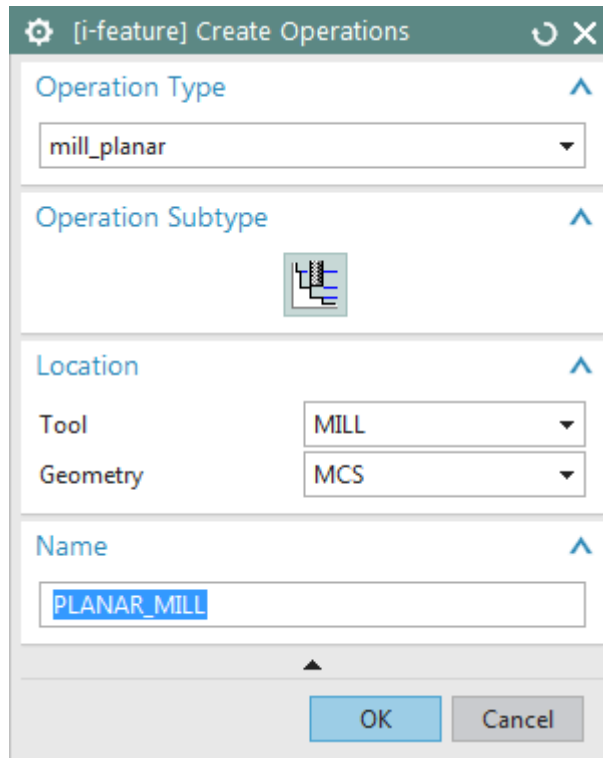


Figure 3.2. The “i-feature Create Operations” window

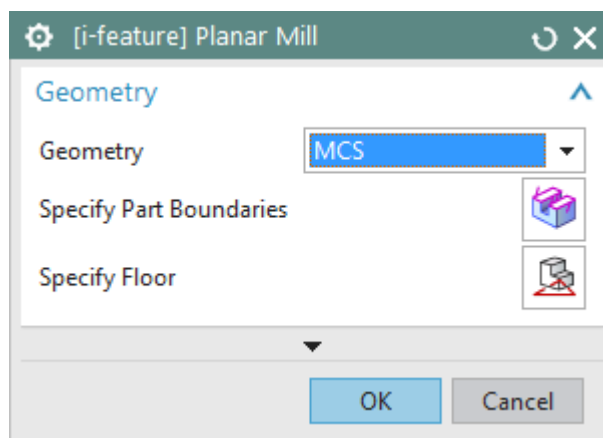


Figure 3.3. The “i-feature Planar Mill” window

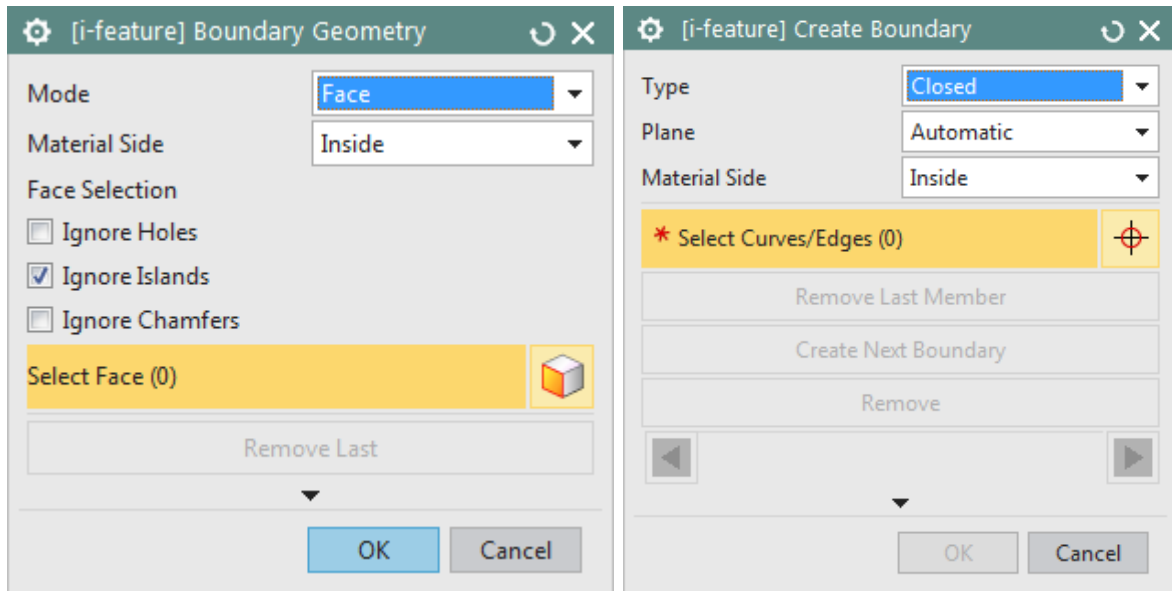


Figure 3.4. The “Boundary Geometry” and “Create Boundary” windows

The “**i-feature Create Operations**” dialog box has an interface similar to that in NX. In the “**i-feature Create Operations**” window it is necessary to choose the type of operation, geometry and tool. The name of operation may be set at your discretion.



If any tool isn’t created yet or a new tool is required for the current operation, set the “**NONE**” parameter. It will be able to create a tool in the “**i-feature Planar Mill**” window.

The “**i-feature Planar Mill**” start screen has two buttons to specify geometry and floor boundaries/parameters. Logic of parameter input is similar to that in NX.



New Siemens NX users will be able to read the detailed description of geometry boundaries/parameters input in Appendix 2.

After successful parameter are specified, the “**i-feature Planar Mill**” window appears as follows in Figure 3.5 (tool parameter is “**NONE**”).

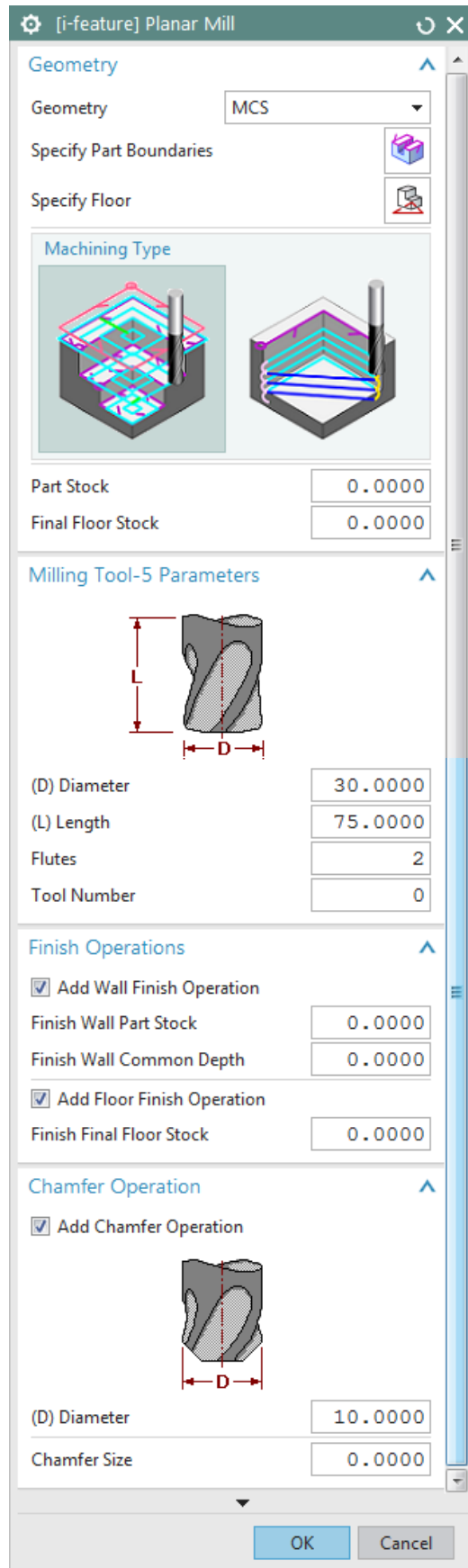


Figure 3.5. The “i-feature Planar Mill” window. All parameters

In the “i-feature Planar Mill” window user can create additional operations and input parameters of the following machining types:

- Rough
- Wall Finish (use the check mark to activate)
- Floor Finish (use the check mark to activate)
- Chamfer Operation (use the check mark to activate)

Rough. Machining type option. After all parameters of the current geometry are determined, machining type option becomes available: “Full” and “Profile” (Figure 3.6).

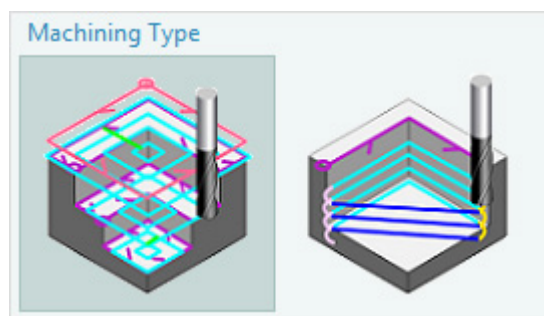


Figure 3.6. Machining type option



The “Full” machining type is used for complete removal of the workpiece material (e.g. workpiece is block)

The “Profile” machining type is used for partial removal of the workpiece material (e.g. workpiece is cast). This option activates the “Additional Passes” field where it is possible to set the number of additional passes for machining of the chosen boundary.



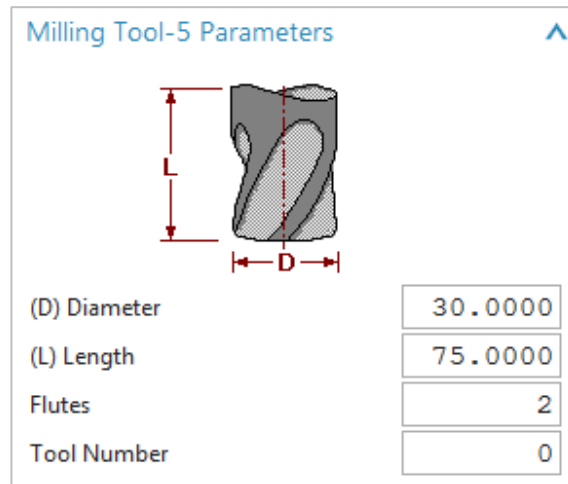
Machining type option is active under certain conditions depending on the chosen geometry and its parameters!

Rough. Part Stocks (Figure 3.7). Two fields in this section allow setting rough stocks for a rough operation (wall stock and floor stock).

| | |
|-------------------|--------|
| Part Stock | 0.0000 |
| Final Floor Stock | 0.0000 |

Figure 3.7. Rough operation part stocks

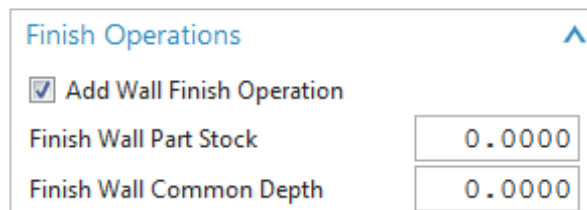
Tool creation. In this section it is necessary to set parameters of the created tool: diameter, length, number of teeth and tool number for postprocessing of operating program (Figure 3.8).



| Milling Tool-5 Parameters | |
|---------------------------|---------|
| (D) Diameter | 30.0000 |
| (L) Length | 75.0000 |
| Flutes | 2 |
| Tool Number | 0 |

Figure 3.8. New tool creation

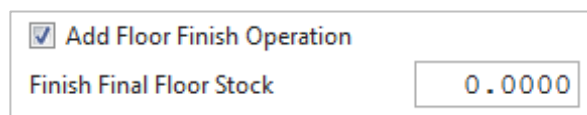
Wall Finish Operation. In this section you can set a finish stock for a wall finish operation and its depth (Figure 3.9).



| Finish Operations | |
|---|--------|
| <input checked="" type="checkbox"/> Add Wall Finish Operation | |
| Finish Wall Part Stock | 0.0000 |
| Finish Wall Common Depth | 0.0000 |

Figure 3.9. Wall Finish Operation

Floor Finish Operation. In this section you can set a finish stock for a floor finish operation (Figure 3.10).



| | |
|--|--------|
| <input checked="" type="checkbox"/> Add Floor Finish Operation | |
| Finish Final Floor Stock | 0.0000 |

Figure 3.10. Floor Finish Operation



Note. When you generate a floor finish operation, parameters of the specified geometry are considered. In some cases, a new finish tool is automatically created. You can change its parameters manually after the generation is over.

Chamfer Operation. In this section it is possible to create a tool for chamfer operation and specify its diameter and chamfer size. Type of the created tool is a chamfering mill with miter shear angle and zero lower radius (Figure 3.11).

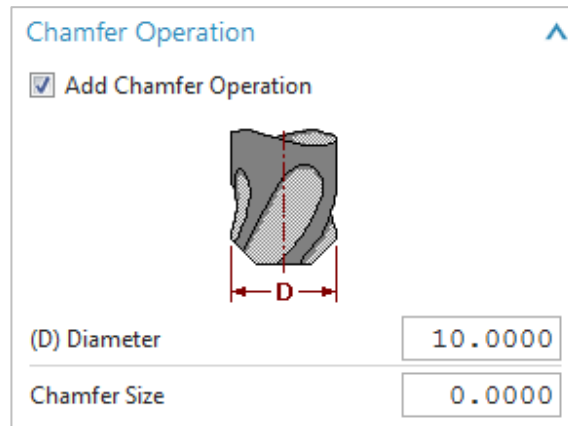


Figure 3.11. Chamfer Operation



All boundaries requiring chamfer must be coplanar!

After all necessary parameters are specified, click “OK”; and then all programs with assigned parameters and generated tool paths will be saved in appropriate directories of program order (Figure 3.12).

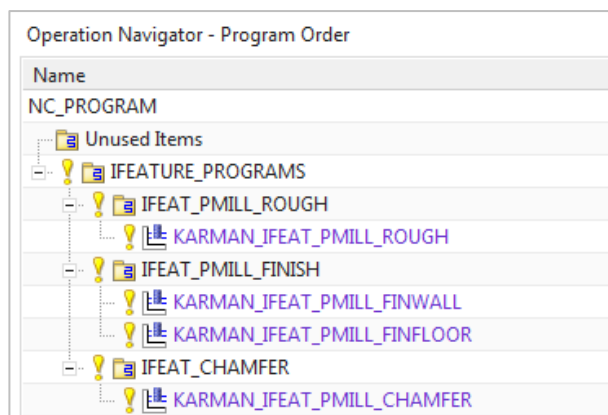


Figure 3.12. Program Order



If necessary, you can correct generated operations with the use of NX common instruments.

4. Uninstalling i-feature

For i-feature uninstallation from your PC, click the **“Uninstall”** shortcut in the **“Start” – “Programs” – “i-feature”** directory. This is the only way to delete i-feature completely from the hard drive and Siemens NX.



To uninstall i-feature, admin rights are also required!

Appendix 1. Detailed description of geometry parameters input in the “i-feature Planar Mill” operations

It is crucial that input of the processed geometry parameters in the “i-feature Planar Mill” operations is very similar to those operations in Siemens NX (such as boundary selection, the location description, parameter setting).

New users, however, may find difficulties in understanding the boundary selection logic and, consequently, may have problems with both i-feature and Siemens NX.

Main combinations of set contours, parameters and suboperations are shown below.

You have to specify the following parameters for tool path generation:

- Boundary type (“Open” or “Closed”)
- Arrangement of material relative to boundary (“Left”, “Right” or “Inside”, “Outside”)
- Boundary (“Face” or “Curves/Edges”)
- Boundary plane (“Automatic” or “User Defined”)
- Floor plane
- Additional options (e.g. ignoring elements in the Boundary “Face” selection)



Activation of the “**Boundary Geometry**” window in the main i-feature window is the start of work with the machining geometry. This window is used to select path and set its parameters (Figure A1.1)

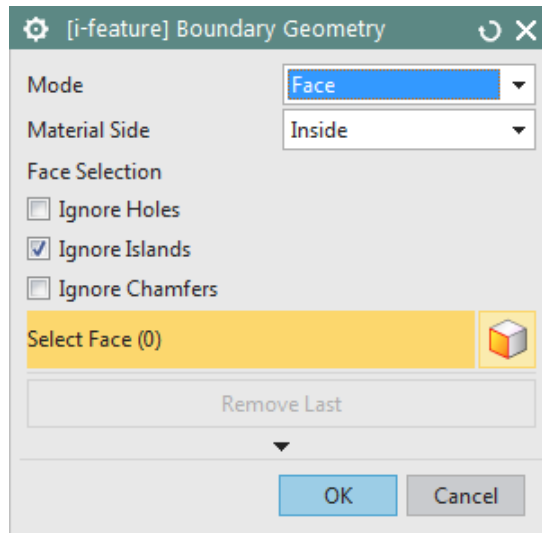


Figure A1.1. The “Boundary Geometry” window

When opening this window, you can immediately specify the geometry by selecting a part face with an indication of “**Material Side**” and parameters of ignoring elements. Let’s consider the example of a functional boundary setting by the “**Face**” mode.

Figure A1.2 shows a machining element; it's the boss arrangement to be machined outside.

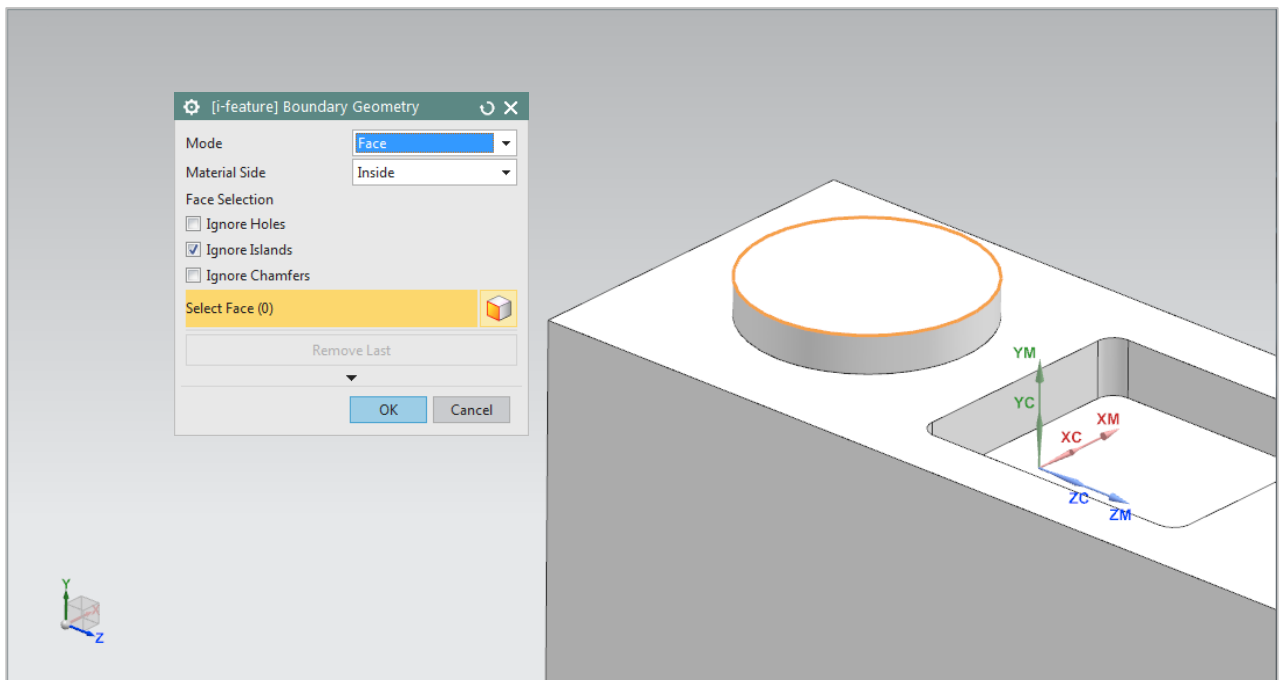


Figure A1.2. Example of the face selection

You should click on the upper face of the boss to select and save the boundary au-

tomatically. Besides, it is considered that the material side is **"Inside"** (that means the metal is inside the boundary and it is necessary to make the boss machining outside), and the parameters of ignoring elements in this case are irrelevant, because the selected boundary has no holes, chamfers and islands inside. Ignoring is used in order not to take these elements as the machining boundary. With this choice, the boundary is automatically assigned to the plane of its location relative to the part. If you want to cancel the current selection, click **"Remove Last member"**. If you want to select several faces, click on them sequentially, and after the selection, click **"OK"** in the **"Boundary Geometry"** window.



Select the **"Material side"** parameter before clicking on the face!

After selecting the boundary geometry and its parameters, you should specify the floor plane in order the system to understand the height of the machining element.



To do so, you need to click the **"Specify Floor"** button and select the surface indicating lower boundary of the machining element. If the surface is not available, you need to create your own floor plane in the constructor of planes

If everything is successfully selected, the i-feature window becomes complete (Figure 3.5), and you can continue to specify the operation parameters and generate the tool path.

The same element can be selected by boundary selection in the **"Curves/Edges"** mode. For this purpose you need to select this mode, and the window view will be changed on that shown in Figure A1.3. In this window there is a field to set the **"Automatic"** or **"Used Defined"** plane mode. Boundary selection is carried out item by item by clicking on the curves. Material side is indicated in the same way as when selecting the boundary through the **"Face"** mode with the only difference being that it is possible to change before clicking the **"OK"** button in the **"Create Boundary"** window.

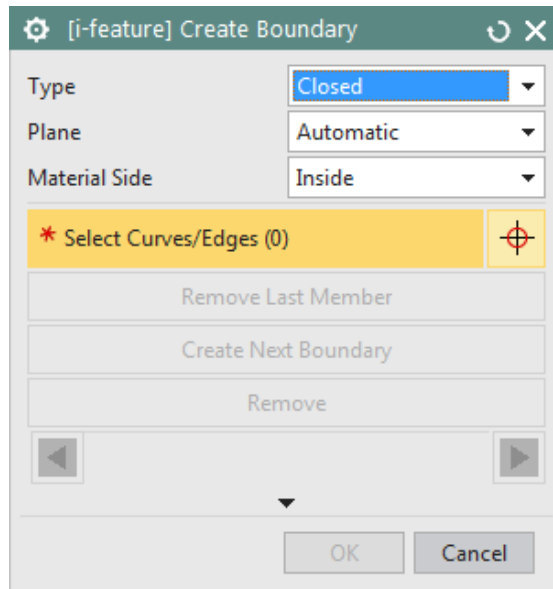


Figure A1.3. The "Create Boundary" window



You can set a plane to the **"Automatic"** mode only if all the selected boundary elements initially lie in one plane. Otherwise, errors may occur!

Sample selection of the **"Closed"** boundary is shown in Figure A1.4. The machining element is a pocket, and all boundary elements lie in one plane, so the **"Automatic"** setting is acceptable. In this example the material side is set to **"Outside"** because removing material is inside the boundary.

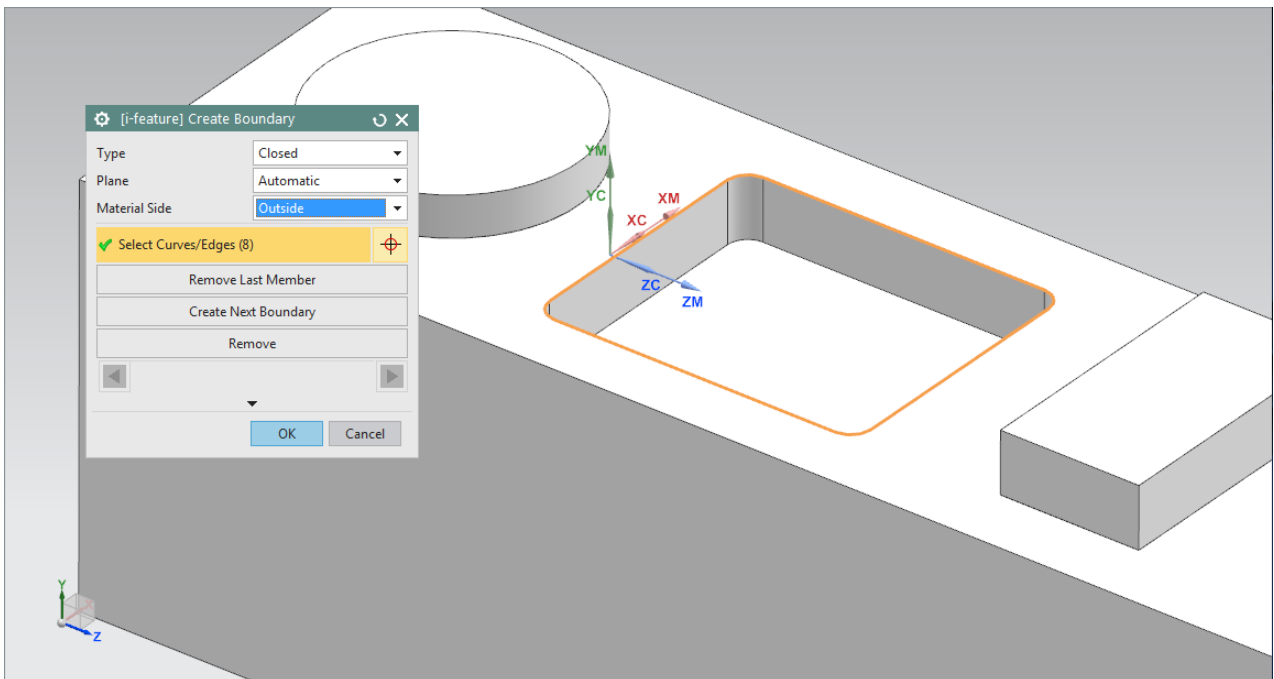


Figure A1.4. Selection of boundary curves

After correct specifying of the boundary and operation parameters, the i-feature system will issue a solution (an example is shown in Figure A1.5).

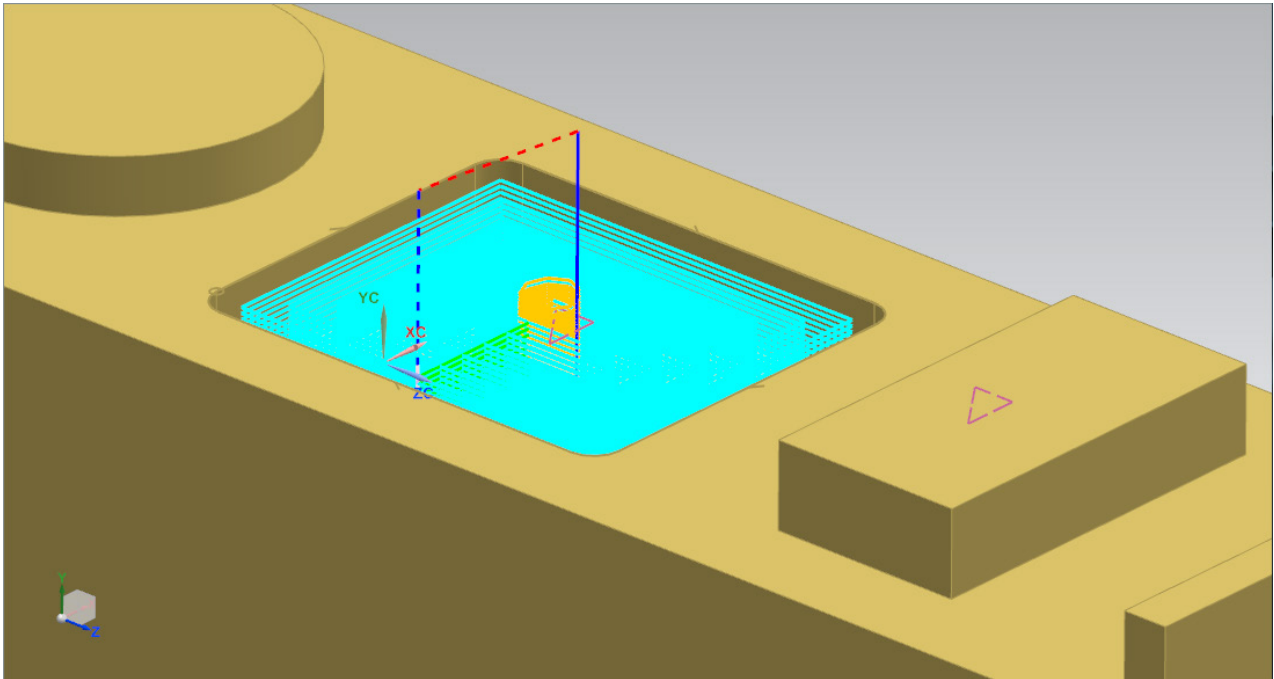


Figure A1.5. Paths generated after setting of all parameters

Consider two examples of the "Open" boundary parameter. In this case, other parameters of material side become accessible: "Left" and "Right".

If you select one of the curve, you should take into account a click point. Figure A1.6 shows selected single curve and material "Left" side.

The red dot indicates a click on the curve. It is located close to the left point of the selected boundary; and the black arrow indicates the direction of future machining. In this case, the "Left" side of the material is selected that corresponds to the actual availability of the material in the illustrated example.



If a click point was closer to the opposite point of the selected curve and material side was also "Left", the machining path would be formed inside the part boss.

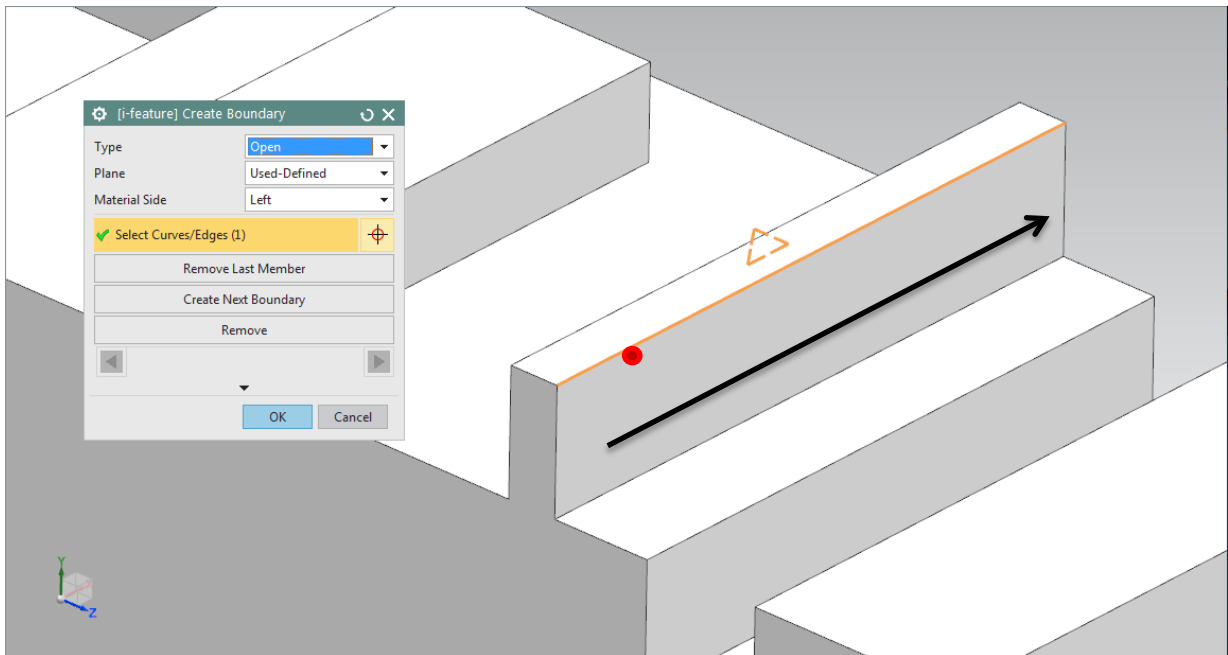


Figure A1.6. Selection of an open boundary and setting of parameters



The **“Automatic”** mode is unacceptable when choosing only one curve of an open boundary!

In the second case, if you select several elements of an open boundary, you should choose them sequentially so that an end point of the first contour coincides with a start point of the next one. It automatically generates the direction shown by the black arrow in the last example. A click point does not matter here.

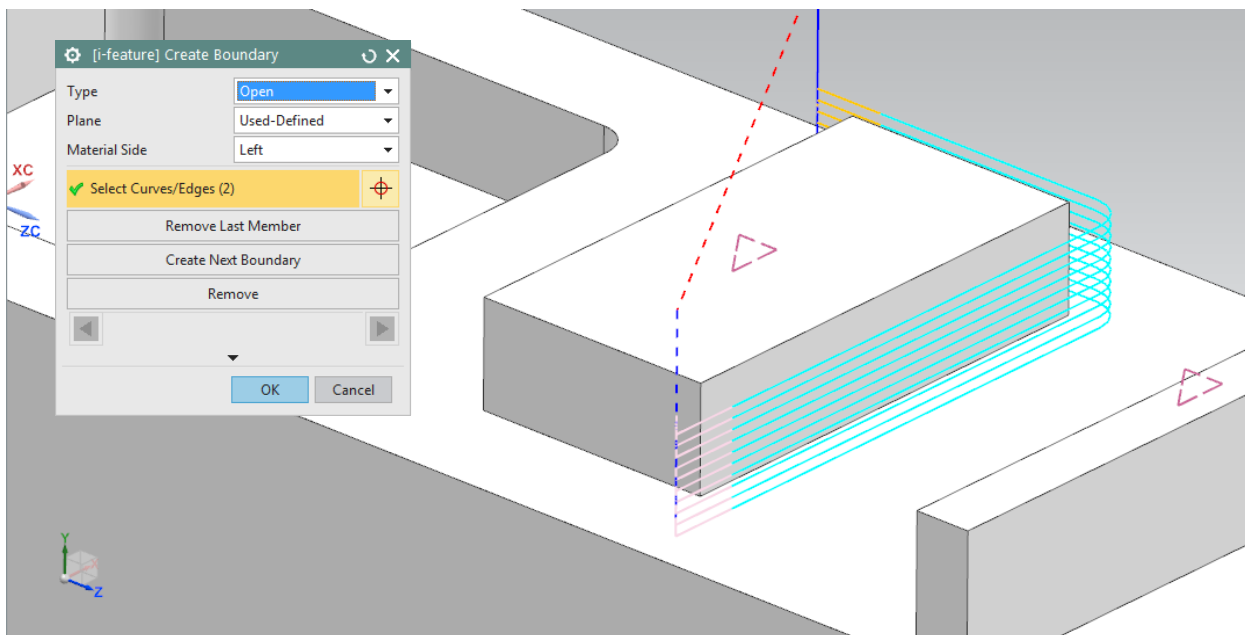


Figure A1.7. Selection of several elements of an open boundary

In Figure A1.7, the described selection and successfully generated path are shown. Two elements are selected (long first, then short), the boundary surface plane and the floor plane at the base of the boss are manually set.

If any boundary element has been selected by mistake, you can cancel it by making the second mouse click while holding down the SHIFT key.

In Figure A1.8, you see how appropriate boundaries can be selected and set:

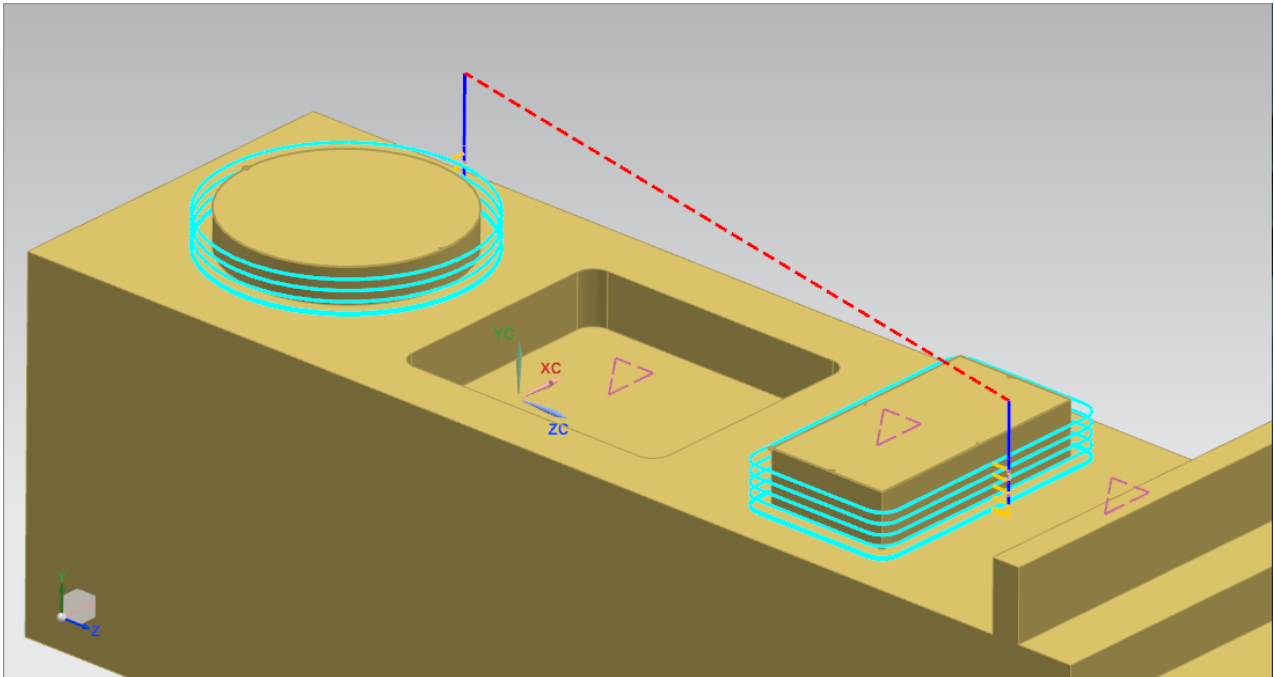


Figure A1.8. Selection of several boundaries

The figure shows that the selected boundaries lie in different planes relative to the detail (different height of bosses). Floor plane is the same for any number of selected boundaries.

In the case of faulty input of the machining geometry or necessary adjustment of its parameters, you need to reenter the “**Create Boundary**” window by clicking the “**Boundary Geometry**” button. In this window, choose the path you created earlier using arrows ◀ ▶ and correct it.